

2012

CFD Modelling in Screw Compressors With Complex Multi Rotor Configurations

Sham Ramchandra Rane
sham.rane.1@city.ac.uk

Ahmed Kovacevic

Madhulika Kethidi

Follow this and additional works at: <https://docs.lib.purdue.edu/icec>

Rane, Sham Ramchandra; Kovacevic, Ahmed; and Kethidi, Madhulika, "CFD Modelling in Screw Compressors With Complex Multi Rotor Configurations" (2012). *International Compressor Engineering Conference*. Paper 2141.
<https://docs.lib.purdue.edu/icec/2141>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at <https://engineering.purdue.edu/Herrick/Events/orderlit.html>

CFD Modeling in Screw Compressors with complex multi rotor configurations

Sham RANE*, Ahmed KOVACEVIC, Madhulika KETHIDI

City University London, Centre for Positive Displacement Compressor Technology,
London, EC1V 0HB, United Kingdom. E-mail: sham.rane.1@city.ac.uk

* Corresponding Author

ABSTRACT

Computational Fluid Dynamics (CFD) of screw compressors is challenging due to the positive displacement nature of the process, existence of very fine fluid leakage paths, coexistence of working fluid and lubricant or coolant, fluid injection and most importantly the lack of methodologies available to generate meshes required for the full three dimensional transient simulations. In this paper, currently available technology of grid remeshing has been used to demonstrate the CFD simulations of complex multi rotor screw compressors like the twin screw compressor with parallel axis and single screw compressor with cylindrical main rotor and two planar gate rotors with perpendicular axis. Presently, methodology for grid generation of constant pitch twin screw machines is available through SCORG© (*Kovacevic et al.*, 2007) but it is currently not suitable for different topologies like that of a single screw or variable pitch rotors. It is very challenging to handle the mesh deformation that happens in the compression chambers during operations of such machines. The methodology tested for this paper uses a technique called key-frame re-meshing in order to supply pre-generated grids to the CFD solver as the solution progresses. In order to evaluate accuracy of such approach, an adiabatic compression-expansion process in a reciprocating piston cylinder arrangement is studied and compared with a diffusion equation based mesh smoothing.

It has been demonstrated, that although it is possible to simulate the complex configuration of screw compressors by key-frame re-meshing technique, there are many limitations with respect to the time consumed in pre-processing, demand to computational resources, accuracy of results and general difficulty to include advanced modeling features like turbulence, multispecies or multiphase flows. Hence it was concluded that customized tools for generation of CFD grids for such complex screw machines still remain to be developed.

1. INTRODUCTION

Rotary Screw Compressors are positive displacement machines. Designs with Twin Screw and Single Screw have been in industrial usage for a long time. The invention of rotary Twin Screw compressor concept dates back to 1878 by Heinrich Krigar in Germany, whereas Single Screw compressors originated around 1962. *Zimmern* (1984) has presented an historical review of the oil-injection free Single Screw Compressors. The design of screw machines is greatly influenced by the dynamics of the working chamber, the suction and the discharge. There have been a number of studies conducted to develop thermodynamic models and use of these models has helped remarkably to evaluate the compressor performance and for optimization of rotor profiles over a period of time. For twin screw compressors with oil free or oil injected operations, *Fujiwara et al.* (1984), *Fukazawa et al.* (1980), *Sangfors* (1982 and 1984), *Singh et al.* (1984 and 1990), *Dagang et al.* (1986), *Kauder et al.* (1994) presented computer models. *Stosic et al.* (1988) developed models by solving numerically the energy and mass differential equations from first principles. *Hanjalic and Stosic* (1997) presented design optimization using such models. *Fleming et al.* (1998) presented model for development and performance improvement of these machines. Recently a book on mathematical modeling of twin screw compressor was published by *Stosic et al.* (2005a). Similarly for Single screw compressors, *Bein and Hamilton* (1982), *Boblitt and Moore* (1984), *Jianhua and Guangxi* (1988), *Hong and Wen* (2004) presented computer model of oil flooded single screw compressors. *Lundberg and Glanvall* (1978) presented similar models and also compared twin and single screw type of compressors at full load. All these studies were performed with an intention of predicting the performance and characteristic of screw machines at the design

stage. With vast improvements in computational technology and availability of more accurate calculation methods, use of Computational Fluid Dynamics for design of the screw machines is encouraged and growing fast. CFD calculations commonly use Finite Volume Method based solvers and its application to screw compressors is an unsteady flow problem with moving boundaries. FVM has been used to solve problems involving unsteady flow with moving boundaries for a long time *Peric* (1985), *Demirdzic* and *Peric* (1990), *Demirdzic* and *Muzaferija* (1995). When FVM is applied to screw compressors, it faces a major challenge in the grids required for transient simulations. As the rotors turn during operation of a compressor, the fluid domain between them is deformed requiring the CFD grid to deform as well. Presently, commercially available general grid generators are not suitable for full three dimensional transient simulations of complex screw compressors. A breakthrough was achieved in 1999 when an analytical rack generation method of *Stosic* (1998) was applied to generate an algebraic, adaptive, block – structured grid calculation for twin screw rotors by *Kovacevic* (1999). Since then there have been several activities reported on CFD analysis of twin screw compressors. *Kovacevic et al.* (1999 and 2000) and *Kovacevic* (2005) have presented the grid generation aspects for twin screw compressors. *Kovacevic et al.* (2002, 2003, 2005b, 2006 and 2007) have reported CFD simulations of twin screw machines for prediction of flow, heat transfer, fluid-structure interaction, etc. *Kovacevic et al.* (2007) also published a textbook on the three dimensional CFD analysis of screw compressors. *Voorde et al.* (2005) have developed a grid conversion algorithm for unstructured to block-structured mesh from solution of Laplace equation for twin screw compressors and pumps. Apart from these works, there are only a few reports available on transient three dimensional CFD analyses of screw machines. All these developments were concentrated on twin screw compressors and so far there are no published works available on single screw machines or other complex screw machines.

In this paper the fundamental aspects of a transient CFD formulation with deforming grids are mentioned. The methodology adopted for this research uses a technique called key-frame re-meshing to supply pre-generated grids to the CFD solver as the solution progresses. Three cases are studied in increasing order of complexity of the working chamber geometry. First one is an adiabatic compression-expansion process in a reciprocating piston cylinder arrangement and here key-frame grid re-meshing methodology is compared with diffusion equation based mesh smoothing to compare the accuracy of results obtained with both the techniques. The second case is a Twin screw compressor where both smoothing and key-frame re-meshing were attempted but the later could not be fully applied due to inability of solvers to handle changes in geometry. However, the customized grid generation algorithm available in SCORG© (*Kovacevic* 1999 and 2000) was feasible. The third case in this research is a Single Screw compressor. However it is not presented in this paper due to lack of space and will be presented in future to further demonstrate use of Key-Frame re-meshing methodology for complex domains of positive displacement machines.

2. FUNDAMENTALS OF A CFD CALCULATION WITH DEFORMING BOUNDARIES

2.1 Laws of conservation and governing equations

In Eulerian reference, conservation of mass, momentum, energy and other intensive properties applied to fluid flow in a control volume (CV) can be defined by coupled, time dependent, partial differential equations. These equations form the basis for all fluid flow computations.

Conservation of Scalar Quantities can be represented by a general transport equation, (*Ferziger* and *Peric*, 1996)

$$\underbrace{\frac{\partial}{\partial t} \int_{\Omega} \rho \phi d\Omega}_{transient} + \underbrace{\int_S \rho \phi \mathbf{v} \cdot \mathbf{n} dS}_{convection} = \underbrace{\int_S \Gamma \text{grad } \phi \cdot \mathbf{n} dS}_{diffusion} + \underbrace{\int_{\Omega} q_{\phi} d\Omega}_{source} \quad (01)$$

Where, Ω is the volume of CV, S is the surface enclosing this CV, \mathbf{n} is the unit vector orthogonal to S and directed outwards, \mathbf{v} is the fluid velocity inside CV in a fixed coordinate system, ρ is fluid density and t is time. Here, the transient term represents rate of change of property ϕ in CV, the convection term accounts for the net advective transport of ϕ across surface S and diffusive transport is described by the gradient. Γ is the diffusivity for the quantity ϕ while q_{ϕ} represents source or sink of ϕ in the CV.

When the control volume is not fixed in space, solution domain changes with time due to movement of the boundaries. This movement is defined either as a function of time or is dependent of the current solution field. The convective fluxes such as the mass flux are calculated in such cases using relative velocity components at the cell faces. If the coordinate system remains fixed and Cartesian velocity components are used, the only change in the

conservation equations is the appearance of relative velocity ($\mathbf{v} - \mathbf{v}_b$) in all convective terms, where \mathbf{v}_b is the velocity vector at the cell face. Application of Leibnitz rule (02) for differentiation under the integral sign to transient terms of equation (01) gives the integral form of general conservation equation in Arbitrary Lagrangian-Eulerian formulation. Arbitrary, because the grid velocity \mathbf{v}_b and grid motion is independent of fluid motion.

$$\frac{\partial}{\partial t} \int_{\Omega} \phi d\Omega = \frac{d}{dt} \int_{\Omega} \phi d\Omega - \int_S \phi \mathbf{v}_b \cdot \mathbf{n} dS \quad (02)$$

$$\underbrace{\frac{d}{dt} \int_{\Omega} \rho \phi d\Omega}_{\text{transient}} + \underbrace{\int_S \rho \phi (\mathbf{v} - \mathbf{v}_b) \cdot \mathbf{n} dS}_{\text{convection}} = \underbrace{\int_S \Gamma \text{grad } \phi \cdot \mathbf{n} dS}_{\text{diffusion}} + \underbrace{\int_{\Omega} q \phi d\Omega}_{\text{source}} \quad (03)$$

However, when the cell faces move, conservation of mass and other conserved quantities is not necessarily ensured if grid velocities are calculated explicitly and in turn are used to calculate the convective fluxes. To ensure that these equations are completely conserved an additional conservation law needs to be satisfied i.e. Space conservation Law.

Space conservation or Geometric conservation is given by,

$$\frac{d}{dt} \int_{\Omega} d\Omega + \int_S \mathbf{v}_b \cdot \mathbf{n} dS = 0 \quad (04)$$

Space conservation can be regarded as mass conservation with zero fluid velocity. The unsteady terms involving integral over control volume Ω , which is now changing with time in governing equations, needs to be treated in a way to be consistent with the space conservation equation with deforming and/or moving grid. Grid velocities and change in CV volumes are required to be calculated and confirming with the space conservation. If the first order backward difference is used for temporal discretization, the transient term can be discretized as:

$$\frac{d}{dt} \int_{\Omega} \rho \phi d\Omega = \frac{(\rho \phi \Omega)^{n+1} - (\rho \phi \Omega)^n}{\Delta t} \quad (05)$$

Where, n and $n + 1$ represent current and next time level respectively. The $n + 1$ volume Ω^{n+1} is computed from

$$\Omega^{n+1} = \Omega^n + \frac{d\Omega}{dt} \Delta t \quad (06)$$

Where, $d\Omega/dt$ is the rate of change of volume of CV and in order to satisfy equation (04) it is calculated as

$$\frac{d\Omega}{dt} = \int_S \mathbf{v}_b \cdot \mathbf{n} dS = \sum_j^{n_f} \mathbf{v}_{b,j} \cdot \mathbf{S}_j \quad (07)$$

n_f is the number of faces on the control volume and \mathbf{S}_j is the j^{th} face area vector. The dot product $\mathbf{v}_{b,j} \cdot \mathbf{S}_j$ on each control volume face is calculated from the volume swept out $-\partial\Omega_j$ by that face over the time step Δt as $\partial\Omega_j/\Delta t$. Therefore mass flux \dot{m}_j can be calculated using $\partial\Omega_j/\Delta t$ instead of explicitly calculated grid velocity $\mathbf{v}_{b,j}$.

$$\dot{m}_j = \int_{S_j} \rho (\mathbf{v} - \mathbf{v}_{b,j}) \cdot \mathbf{n} dS_j \approx \rho_j (\mathbf{v} \cdot \mathbf{n})_j S_j - \frac{\rho_j \partial\Omega_j}{\Delta t} \quad (08)$$

If the volume change and mass fluxes are calculated as above, space conservation is conformed. The requirement of space conservation in flow equations on moving integration points was introduced by (Trulio and Trigger, 1961) and Thomas and Lombard (1979). The importance of the space conservation law was discussed by Demirdzic and Peric (1988). They showed that error in mass conservation due to nonconformance of space conservation is proportional to the time step size for constant grid velocities and is not influenced by the grid refinement size.

Theoretically, when the equations of conservation are integrated over control volumes of infinitesimal size they can completely resolve the flow dynamics. But when applied to control volumes of finite dimensions there are limitations in terms of resolving length scales beyond certain size or capturing near wall boundary layer phenomenon or shocks where the gradients are high. So usually in addition to these equations there can be more models introduced into the calculation like turbulence models, near wall functions, Species transport etc. These models can be analogously represented by the general transport equation and are not discussed here.

It is also possible to change the grid topology from one time step to another (Ferziger and Peric, 1996) since the computation of surface and volume integrals is not dependent on solution from previous time steps. This concept has been utilized here to solve for complex domain deformations of screw compressors.

2.2 Solution of governing equations

The governing equations form a closely coupled, time dependent set of PDE's and commonly a Finite Volume Method for solution of these equations is employed. The model geometry is spatially discretized into a number of control volumes and the equations of conservation are integrated over all of them. Closed loop solutions to the governing equations are available only for very few limited ideal geometries and it is impractical to be obtained for realistic machines. Therefore, it is a practice to adopt an iterative, time advancing procedure for solving such systems. The presence of a deforming domain further complicates the calculations imposing practical restrictions on time step size or grid size used for discretization (Ferziger and Peric, 1996).

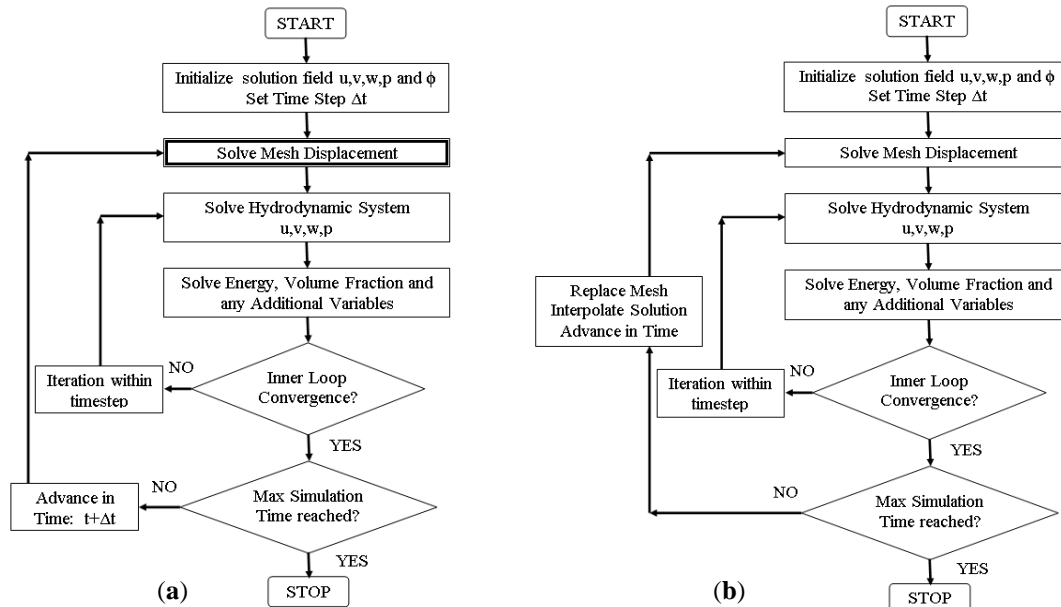


Figure 1: Flow Chart of Solution Process with Deforming Domains (a) Mesh Smoothing and (b) Key-frame re-meshing

Figure 1a represents a flow chart of the solution process for FVM with deforming domains. Here the highlighted block of solving mesh displacement is of particular interest. This mesh displacement moves the initial grid to the new position in time and control volume deforms but there is no change in the number of cells or their connectivity during this step. This is followed by integration and solution of governing equations, conforming to the space conservation law on the deformed mesh at this new time level. After the time step is converged the solution advances to next time step and next solution of mesh displacement is performed. No intermediate interpolation is required as necessary data is available for all cells within the domain. Figure 1b represents a flow chart of the solution process that can be used for high deformation and applications like screw compressors. The process is called Key-Frame re-meshing method. Here, after every time step, a check is made on the control volume cell quality or possibility to absorb further deformation without generating negative element volumes and selectively some group of cells are locally remeshed. This changes the number of cells and their connectivity from one time step to the other. The remeshing is followed by next time step calculations similar to Figure 1a. An intermediate interpolation is required as all cells have changed connectivity. In this paper an attempt has been made to use the algorithm in Figure 1b by supplying to the solver, complete deforming domain grids on which the corresponding time level data gets interpolated. The process continues until maximum simulation time is reached. Depending on the type of solvers an inner coefficient loop over all cells needs to be converged first.

3. METHODOLOGY USED FOR GRID DEFORMATION

There are two methods of solving CFD problems with deforming meshes. In the first case, the mesh deforms depending on solved variables as for example –in case of deformation of the compressor rotors due to fluid pressure (Kovacevic *et al*, 2002). Alternatively, the grid deformation may be known and pre-specified as a function of time. This is the case of the working chamber of a screw compressor in which the rotor displacement is known as a function of time.

In the present work, a methodology of Key-Frame re-meshing as shown in Figure 1b was evaluated. It consists of generating a set of numerical meshes required by the solver during every time step in advance of the CFD calculations. These meshes are then passed to the solver cyclically during calculations to allow smooth transition within the simulation. This set of mesh was generated using general purpose grid generators. During the solution process, an interrupt condition was provided to the solver by means of a function which directed the solver to replace the mesh for every time step. This interrupt condition called a user defined 'Perl' script was used to run the pre-processor in batch mode and to replace the deformed grid in the current time step with the new grid at same time step. It then writes the new solver input file. Once this batch process is completed, the solver reads the new input file, interpolates previous results onto the new mesh and marches to the next time step which then accounts for mesh deformation. Effectively this process represents a grid re-meshing performed external to the solver.

Advantages of this method are namely, i) Commercial unstructured meshing tools may be utilized to generate the set of meshes required for every time step, ii) Depending on the geometry it may be possible to generate only a small number of meshes for different position of geometry and use them cyclically to complete a compressor cycle. For example in a 6 lobed main rotor single screw compressor, each lobe will occupy 360/6 degrees. By generating numerical mesh for every 1 degree, only 60 meshes are required to cover the full cycle, iii) More control on mesh quality can be achieved.

There are certain limitations of such approach, namely i) Geometry of a screw compressor with very complex leakage paths may be very difficult to obtain the exact representation and a very fine mesh is required, ii) It is required to accurately capture the geometry changes in the leakage region, happening with every timestep, iii) Automation of the pre-processing is difficult as geometry as well as meshing has to be repeated for every time step. Hence application of this methodology is a time consuming process.

The diffusion equation based smoothing, as presented in Figure 1a uses different approach. In this method the specified displacement on boundary nodes is diffused to interior nodes by solving equation (09).

$$\nabla \cdot (\Gamma_{disp} \nabla \delta) = 0 \quad (09)$$

Where, δ is the nodal displacement and Γ_{disp} is the mesh stiffness that depends on local cell volumes and distance of the nodes from deforming boundaries. There is no change in the connectivity and count of the mesh and grid is deformed at the beginning of every time step. This method is not suitable for large displacements.

4. CASE STUDIES

From the discussion in section 3 it has been found that for transient CFD analysis of screw compressors, the methodology of Key-Frame based remeshing can be adopted to certain extent. Three cases were studied to help in arriving at conclusions to the selection of the key-frame remeshing method for screw compressor simulations. In the First case the reversible adiabatic compression process in a piston cylinder was solved by use of both methods listed above. This case forms the fundamental mechanism of a positive displacement compressor. The second case was a single screw compressor with a realistic complex topology with the main zone of deformation embedded in main rotor and third case was a twin screw compressor with main zone of deformation embedded in both male and gate rotors. For the sake of space, in this paper only two of these case studies are presented.

4.1 Simulation of a simple Piston Cylinder Configuration

The purpose of this analysis is to compare the results from Diffusion Smoothing based Mesh Motion and Key-Frame based remeshing method with the theoretical results. It is expected that Diffusion Smoothing will be exact as there are no interpolation errors from one time step to the other. Consider a piston cylinder arrangement with a sinusoidal displacement given to the piston. The process can be modeled by polytropic process equation relating the gas pressure and volume. Gas in the cylinder gets compressed when there is reduction in volume and its pressure and temperature increases due to the work done. Conversely during expansion, the work is extracted and pressure and temperature decrease. Since the process adiabatic and reversible there will be no energy losses or gains in the control volume and the gas will return to its initial state.

$$\left(\frac{p_2}{p_1}\right) = \left(\frac{V_1}{V_2}\right)^\gamma = \left(\frac{T_2}{T_1}\right)^{\frac{\gamma}{\gamma-1}} \quad (10)$$

Where, p is Absolute Pressure, V is specific Volume and T is temperature. Subscripts 1 and 2 denote the initial and final states respectively. For this trial, a cylinder with diameter 100.00 mm and length 100.00 mm was considered.

The initial position of the piston is at the maximum volume of 10000.00 mm^3 . The Piston displacement is 70.00 mm varying sinusoidally with 50.00Hz frequency. The final minimum cylinder volume is 3000.00 mm^3 . This gives a fixed volume ratio of 3.33 for the system. Based on equation (10), for an initial cylinder absolute pressure of 2.01 bar the expected peak pressure is 10.86 bar . Similarly, for an initial temperature of 298.00 K the expected peak temperature is 482.36 K .

The Hexahedral mesh is the most suitable for diffusion smoothing as the cell quality does not deteriorate considerably when boundaries deform. Figure 2b shows the mesh at three different time steps generated for diffusion smoothing. The cell connectivity and the node count does not change during the calculation from one time step to the next. For the Key-Frame based remeshing a tetrahedral mesh is selected - Figure 2c. since in the real screw compressor the geometry is complex and hexahedral meshes are difficult to be generated. This represents the closed system in which all boundaries are modeled as walls. Only the piston is specified with a displacement as a function of time. The time step size of $2.8571\text{e}^{-04} \text{ sec}$ was used with the implicit second order backward Euler discretization. The pressure based coupled solver is used. Advection scheme was high resolution and turbulence model was k-epsilon. An r.m.s. residual target of 1.0e^{-04} was maintained for all equations. The used gas was air regarded as ideal gas with the molar mass of 28.96 kg/kmol , Specific Heat Capacity $1.0044\text{e}^3 \text{ J/kg K}$, Dynamic Viscosity $1.831\text{e}^{-05} \text{ kg/m s}$ and Thermal Conductivity $2.61\text{e}^{-02} \text{ W/m K}$.

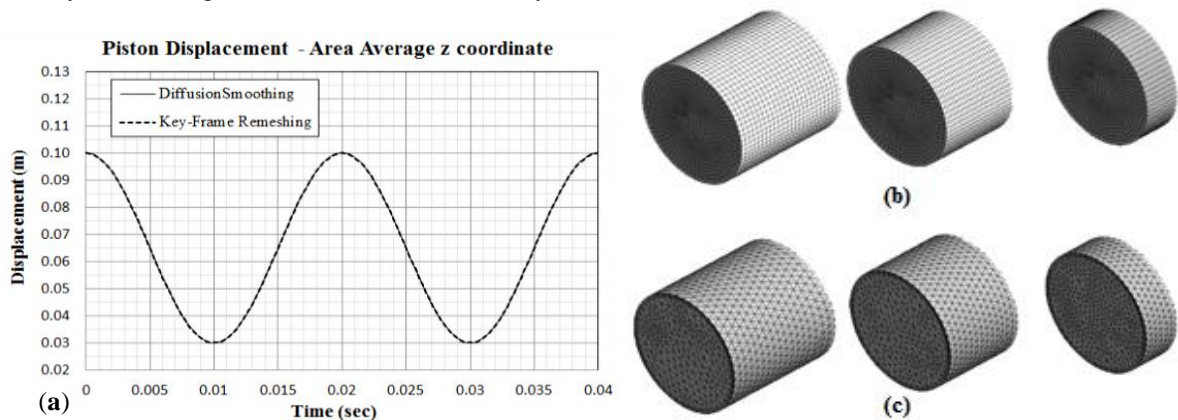


Figure 2: (a) Comparison of Piston Displacement with Diffusion Smoothing and Key-Frame remeshing, (b) Hexahedral Mesh using Diffusion Smoothing, (c) Tetrahedral Mesh used in Key-Frame remeshing

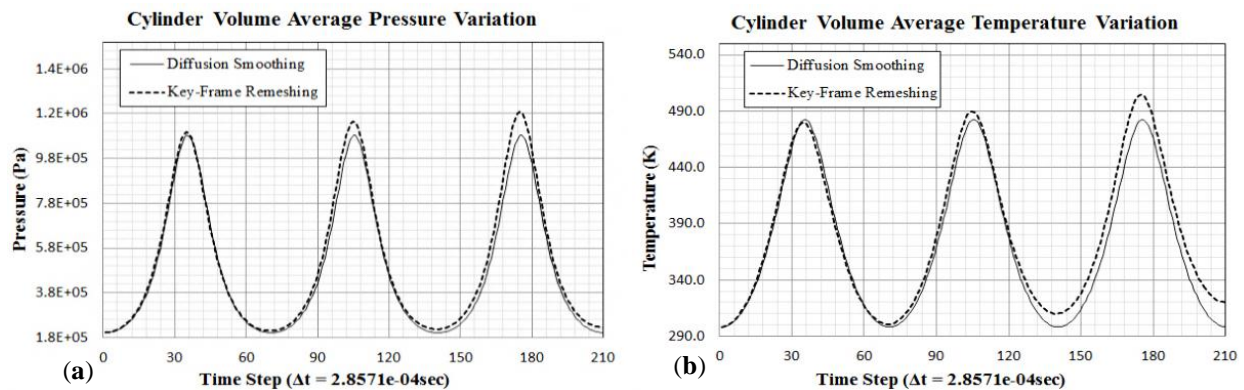


Figure 3: Comparison of (a) Pressure and (b) Temperature, with Diffusion Smoothing and Key-Frame remeshing

Figure 2a shows the piston displacement with time for both cases showing the same volume ratio in both cases. Figure 3a shows the change of pressure with time in both cases. Although both cases have the same volumetric compression and expansion the achieved peak pressures are not equal. In case of diffusion smoothing the pressure in the first cycle achieves the theoretical peak of 10.86 bar and consistently repeats itself in the following cycles. But in the case of Key-Frame remeshing the maximum pressure in the first cycle is little higher than the theoretical and in the following cycles it continues to increase. Similarly the initial state of pressure at the end of expansion does not return to its base value as it does in case of diffusion smoothing.

Table 1: % error in calculation of Pressure and Temperature

Quantity	% error					
	Ist Cycle		IInd Cycle		IIIrd Cycle	
	Case-1	Case-2	Case-1	Case-2	Case-1	Case-2
Absolute Pressure	-0.44	1.15	-0.42	5.38	-0.41	9.33
Static Temperature	-0.08	-0.53	-0.07	1.54	-0.05	4.57
Case-1 Diffusion Smoothing, Case-2 Key-Frame Remeshing						

Figure 3b also shows the temperature change with time in the cylinder for both cases. It is seen that peak temperatures are not equal in the two cases. In case of diffusion smoothing the temperature in the first cycle goes to the theoretical peak of 482.35 K and repeats itself in the following cycles consistently. In the case of Key-Frame remeshing the peak temperature in the first cycle is equal to the theoretical but in the following cycles it continues to increase. Similarly the initial state of temperature, at the end of expansion does not return to its base level of 298.00 K as it does in case of diffusion smoothing.

Table 1 shows the percentage error in the prediction of pressure and temperature by both methods and also its variation over multiple consecutive cycles. These results show that diffusion smoothing based method is highly accurate and conforms to the theoretically expected results for a deforming boundary formulation. But there are errors in the pressure and temperature prediction using the key-frame remeshing based method where the mesh is replaced after every time step. These errors can be attributed to the interpolation of results from one time step to the other on the replaced mesh. There could also be some violation of space conservation equation happening as the mesh is replaced every time step. As pointed out in *Ferziger and Peric* (1996), this can lead to artificial mass source errors in the continuity equation that can also accumulate with flow time. Limiting the mesh to be replaced only when cell quality goes bad should help in reducing the error but in case of complex topologies like the screw compressors this is very difficult. This analysis gave important information about the level of accuracy to be expected with the key-frame remeshing approach.

4.2 Simulation of a Twin Screw Compressor Configuration

For the CFD analysis of twin screw machines, the grid generation tool called SCORG© is available (*Kovacevic et al.*, 2007). Similarly, unstructured grids can be manipulated to generate block structured grids as shown by (*Voorde J et al.* 2005). Still an attempt was made to use general purpose grid generators since neither of the above methods could handle variable pitch geometries or geometries with non-parallel axes. The intention was to solve a 3/5 combination, oil free twin screw compressor with the male rotor diameter $\Phi 127\text{mm}$. Initially, an attempt was made to generate a complete model that included both radial and interlobe leakage gaps as shown in Figure 4a. A very fine grid was generated to capture the clearances. It was found that for a male rotor speed of 7000rpm, the max displacement that could be given to the rotor was about 0.1 degree per time step before the formation of negative element volumes occurs. Therefore it was not practically possible to continue with this calculation. The second attempt was carried out to exclude the radial clearances. This further complicated the geometry as shown in Figure 4b after which it was concluded that the key-frame re-meshing could not be applied to the twin screw compressor geometry without an excessive computing resources and time. In order to demonstrate the applicability of a customized grid generator, SCORG© (*Kovacevic et al.*, 2007) was used and a block structured hexahedral mesh was generated.

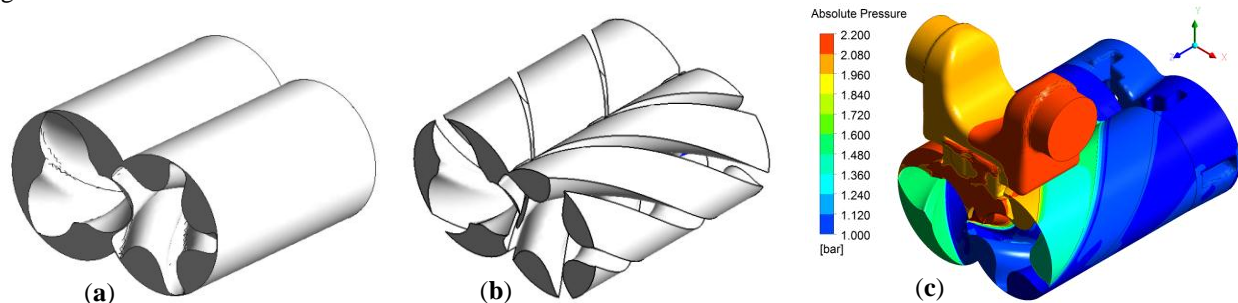


Figure 4: Deforming rotor domain (a) Including Leakage Clearances (b) Radial Leakage Excluded (c) Contours of Pressure variation

Figure 4c shows the contours of Absolute Pressure for a pressure ratio of 2.0 bar. Kovacevic *et al.*, (2002, 2003, 2005b, 2006 and 2007) have reported CFD simulations of twin screw machines for prediction of flow, heat transfer, fluid-structure interaction, etc., using the same grid generator tools. Since only grid smoothing is used and the mesh connectivity does not change, there is no intermediate interpolation. This gives a high accuracy to the calculations as seen from the piston-cylinder case. Also the grid generators are very robust and quick. This remarkably reduces the time consumed in pre-processing, that was experienced to be too high in the single screw and twin screw compressor cases where key-frame remeshing was used. Typically the pre-processing time required when using the key-frame re-meshing methodology is around 50 hours machine time whereas when using SCORG© it is not more than 2 hours machine time. Importantly, leakages in the radial and interlobe clearances are captured. This case study shows a possible limitation to the application of key-frame remeshing to solve complex screw compressor problems.

5. CONCLUSION

In this paper, the methodology of key frame re-meshing is applied to solve flow in complex geometry of screw compressors. The working chamber of the compressor at different time steps is represented by a set of grids which are passed to the solver at appropriate time steps thus avoiding failure of mesh due to large deformation. Successful simulations of a Piston-Cylinder and a Single Screw compressor were carried out. All attempts to solve flow within twin screw compressor failed due to complexity of numerical mesh. Important parameters like pressure or temperature vs. angle diagrams, flow rates, compression power, regions of dynamic losses and oscillations could be identified in solved cases. This case study shows that though it is possible to simulate some of the complex configurations of screw compressors by using general purpose grid generators, there are many limitations. Time consumption for pre-processing, lack of accuracy, inability to include leakages, and limitations in complex domains such as that of a twin screw compressor are obvious ones. The need still exists to develop customized tools to generate CFD grids for complex screw machines such as single screw, variable pitch machines, etc. The case study of Twin screw compressor confirms that methodology in SCORG© with customized tools and techniques, can be further developed and adopted in an industrial or research environment to serve as main simulation methodology for the screw compressors.

NOMENCLATURE

CV	control volume	Ω	Volume of CV
S	Surface enclosing CV	\mathbf{n}	Unit vector orthogonal to S directed outwards
\mathbf{v}	Fluid velocity inside CV in a fixed coordinate system	ρ	Fluid density
t	Time	ϕ	Scalar transported in CV
Γ	Diffusivity for the quantity ϕ	q_ϕ	Source or Sink of ϕ in the CV
δ	Nodal displacement	Γ_{disp}	Mesh stiffness
p	Absolute Pressure	V	Specific Volume
T	Temperature	Subscripts 1 and 2 denote fluid initial and final state	

REFERENCES

- Boblitt W. W. and Moore J., 1984, Computer Modeling of Single -Screw Oil Flooded Refrigerant Compressors, *Proc. Int. Compressor Conf. at Purdue*, Paper 506.
- Dagang X., Xion Z., Yu Y., 1986, The Computer Simulation of Oil Flooded Refrigeration Twin-Screw Compressor, *Proc. Int. Compressor Conf. at Purdue*, Paper 345
- Demirdzic I., Muzafaric S., 1995, Numerical Method for Coupled Fluid Flow, Heat Transfer and Stress Analysis Using Unstructured Moving Mesh with Cells of Arbitrary Topology, *Comp. Methods Appl. Mech. Eng.*, Vol.125, pp. 235-255
- Demirdzic I., Peric M., 1990, Finite Volume Method for Prediction of Fluid flow in Arbitrary Shaped Domains with Moving Boundaries, *Int. J. Numerical Methods in Fluids* Vol.10, p. 771
- Ferziger J. H. and Peric M., 1996, *Computational Methods for Fluid Dynamics*, ISBN 978-3-540-42074-3, Springer, Berlin, Germany.
- Fleming J. S., Tang Y., Cook G., 1998, The Twin Helical Screw Compressor, Part 1: Development, Applications and Competitive Position, Part 2: A Mathematical Model of the Working process, *IMEchE, Journal of Mechanical Engineering Science*, Vol. 212, p. 369.

- Fujiwara, M., Kasuya, K., Matsunaga, T. and Watanabe, M., 1984, Computer Modeling for Performance Analysis of Rotary Screw Compressor. *Proc. Int. Compressor Conf. at Purdue*, Paper 503.
- Hanjalic K., Stosic N., 1997, Development and Optimization of Screw machines with a simulation Model – Part II: Thermodynamic Performance Simulation and Design Optimization. *Transactions of the ASME* 664 / Vol. 119
- Jianhua Wu and Guangxi Jin, 1988, The Computer Simulation of Oil-Flooded Single Screw Compressors, *Proc. Int. Compressor Conf. at Purdue*, Paper 646.
- Kauder K., Rau B., 1994, Auslegungsverfahren für Schraubenkompressoren. (Design Procedure for screw compressors), VDI Berichte 1135, S. 31-44, Düsseldorf: VDI-Verlag.
- Kovacevic A., Stosic N., Smith I. K., 1999, Development of CAD-CFD Interface for Screw Compressor Design, *International Conference on Compressors and Their Systems, London, IMechE Proceedings*, 757
- Kovacevic A., Stosic N., Smith I. K., 2000, Grid Aspects of Screw Compressor Flow Calculations, *ASME Congress*, Orlando FL
- Kovacevic A., 2005, Boundary Adaptation in Grid Generation for CFD Analysis of Screw Compressors, *International Journal for Numerical Methods in Engineering (IJNME)*, vol. 63.
- Kovacevic A., Stosic N. and Smith I. K., 2002, Numerical Simulation of Fluid Flow and Solid Structure in Screw Compressors, *Proceedings of ASME Congress*, New Orleans, IMECE2002-33367.
- Kovacevic A., Stosic N. and Smith I. K., 2003, Three Dimensional Numerical Analysis of Screw Compressor Performance, *Journal of Computer Methods in Applied Mechanics and Engineering*.
- Kovacevic A., Stosic N., Smith I. K., 2006, Numerical simulation of combined screw compressor–expander machines for use in high pressure refrigeration systems, *Simulation Modeling Practice and Theory*, Volume 14, Issue 8, Pages 1143–115
- Kovacevic A., Stosic N. and Smith I. K., 2007, *Screw compressors - Three dimensional computational fluid dynamics and solid fluid interaction*, ISBN 3-540-36302-5, Springer-Verlag Berlin Heidelberg New York.
- Kovacevic A., Stosic N. and Smith I. K., 2002, Influence of Rotor Deflection upon the Screw Compressor Process, *Schrauben compressor tagung*, Dortmund.
- Li Hong Qi and Jin Li Wen, 2004, Design Optimization of an Oil-Flooded Refrigeration Single Screw Compressor, *Proc. Int. Compressor Conf. at Purdue*, Paper 1715.
- Lundberg A. and Glanvall R., 1978, A Comparison of SRM and Globoid Type Screw Compressors at Full Load, *Proc. Int. Compressor Conf. at Purdue*, Paper 288.
- Peric, M., 1985, A Finite Volume Method for the Prediction of Three Dimensional Fluid Flow in Complex Ducts, *PhD Thesis, Imperial College of Science, Technology & Medicine*, London
- Singh P. J., Onusach A. D., 1984, A Comprehensive Computerized Method for Twin Screw Rotor Profile Generation and Analysis, *Proc. Int. Compressor Conf. at Purdue*.
- Stosic N., Hanjalic K., Kovacevic A. and Milutinovic Lj, 1988, Mathematical Modeling of the Influence of Oil on the Working Cycle of Screw Compressors. *Proc. Int. Compressor Conf. at Purdue*
- Sangfors B., 1984, Computer Simulation of the Oil Injected Twin Screw Compressor, *Proc. Int. Compressor Conf. at Purdue*, 528
- Stosic N., Smith I.K. and Kovacevic A., 2005a, *Screw Compressors: Mathematical Modeling and Performance Calculation*, Monograph, Springer Verlag, Berlin, June 2005, ISBN: 3-540-24275-9
- Stosic N., Smith I. K. and Kovacevic A., 2005b, Numerical Investigation of Heat Transfer on Screw Compressor Rotors. *8th Argentinian Congress on Computational Mechanics*, Buenos Aires, Argentina.
- Stosic N., 1998, On Gearing of Helical Screw Compressor Rotors, *Proceeding of IMechE, J. Mech. Eng. Science*, Vol.212, and pp. 587.
- Thomas P. and Lombard C., 1979, Geometric conservation law and its application to flow computations on moving grids, *AIAA*, 17, 1030–1037.
- Thomas W. Bein, James F Hamilton, 1982, Computer modeling of an oil flooded Single Screw Compressor. *Proc. Int. Compressor Conf. at Purdue*, Paper 383.
- Voorde John Vande and Vierendeels Jan, 2005, A grid manipulation algorithm for ALE calculations in screw compressors. 17th AIAA Computational Fluid Dynamics Conference, Canada, AIAA 2005-4701.
- Wei-feng WU, Quan-ke FENG, 2008, Leakage Paths of Two Type of Meshing Pairs in Single Screw Compressors, *Proc. Int. Compressor Conf. at Purdue*, Paper 1864.
- Zimmern Bernard, 1984, From Water to Refrigerant: Twenty Years to Develop the Oil Injection-Free Single Screw Compressor. *Proc. Int. Compressor Conf. at Purdue*, Paper 500.
- Zimmern B., Patel Ganshyam, 1972, Design and Operating Characteristics of the Zimmern Single Screw Compressor, *Proc. Int. Compressor Conf. at Purdue*, Paper 16.